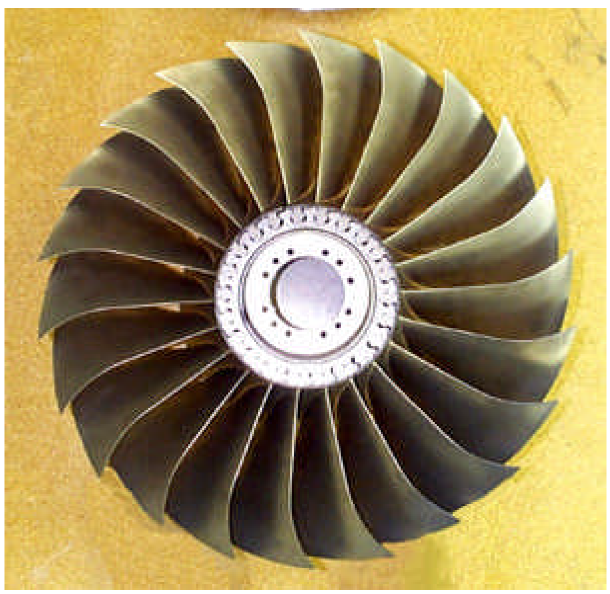
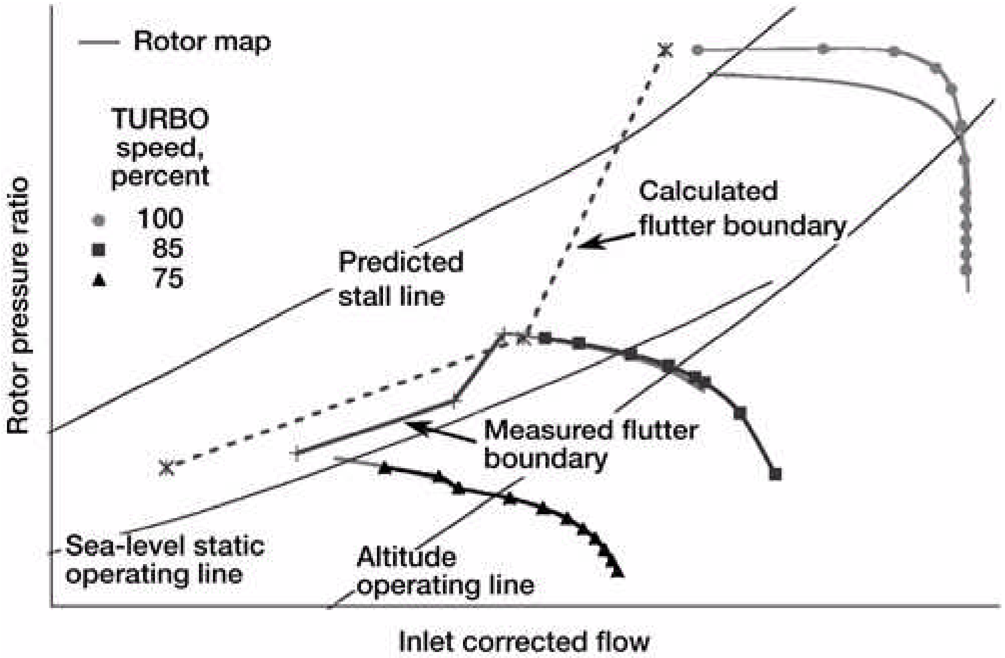
Forward-Swept Fan Flutter Calculated Using TURBO Code

Flutter, a self-excited dynamic instability arising because of fluid structure interaction, can be a significant design problem for rotor blades in gas turbines. Blade shapes influenced by noise-reduction requirements increase the likelihood of flutter in modern blade designs. Validated numerical methods provide designers an invaluable tool to calculate and avoid the flutter instability during the design phase. Toward this objective, a flutter analysis code, TURBO, was developed and validated by researchers from the NASA Glenn Research Center and other researchers working under grants and contracts with Glenn. The TURBO code, which is based on unsteady three-dimensional Reynolds-averaged Navier-Stokes equations was used to calculate the observed flutter of a forward-swept fan. The forward-swept experimental fan, designed to reduce noise, showed flutter at part-speed conditions during wind tunnel tests.



Forward-swept fan.

The forward-swept experimental fan shown in the photograph consisted of 22 forward-swept inserted blades. The steady performance of the fan was mapped first at three different speed lines. Then, we calculated flutter by calculating the work done by the fluid on the rotor blades for the three speed lines. The blades were prescribed a harmonic motion at the natural frequency, mode shape, and nodal diameter of interest. The work done on the blade by the fluid over one cycle of vibration was converted to a more meaningful damping value referred to as aerodynamic damping. A negative aerodynamic damping implies a dynamically unstable blade, or blade with flutter. The aerodynamic damping was calculated at a minimum of two operating points on a given speed line. The flutter point was then calculated as the mass flow point where the aerodynamic damping value went to zero.



Comparison of calculated flutter boundary and performance for the forward-swept fan with experimental data.

Long description of figure 2. Graph of rotor pressure ratio versus inlet correct flow for TURBO speeds of 100, 85, and 75 percent, showing rotor map, sea-level static operating line, altitude operating line, measured flutter boundary, predicted stall line, and calculated flutter boundary.

The results obtained for three different speed lines are shown in the graph. The steady performance and the calculated and observed flutter points are also shown in this figure. The flutter was observed for a two-nodal-diameter forward traveling wave in the first natural mode. The calculated flutter point also corresponds to the first natural mode and the two-nodal-diameter pattern. Very good correlation was found between the analysis and measurements. The code correctly identified the observed flutter and the characteristics of the flutter. Several parametric studies were also carried out to better understand the flutter behavior. During the study, we found that the effect of variations in inflow and exit boundary conditions, tip gap, and vibration amplitude was limited and did not strongly affect the calculated flutter point. We also found that for operating conditions where flutter was dictated by the presence of a shock wave, an inviscid analysis gave results qualitatively similar to viscous analysis. Thus, the inviscid analysis could be used successfully in a design environment for screening the designs. A more rigorous viscous analysis could then be calculated at critical points identified by the inviscid analysis. Both the viscous and the inviscid analyses with the TURBO code are currently being used to redesign the forward-swept fan to be flutter free.

University of Toledo contacts: Dr. Rakesh Srivastava, 216-433-6045, Rakesh.Srivastava@grc.nasa.gov; and Dr. Milind A. Bakhle, 216-433-6037, Milind.A.Bakhle@grc.nasa.gov
Glenn contact: George L. Stefko, 216-433-3920, George.L.Stefko@nasa.gov

Authors: Dr. Rakesh Srivastava, Dr. Milind A. Bakhle, and George L. Stefko
Headquarters program office: OAT
Programs/Projects: QAT